ICHTFM 2018 : 20th International Conference on Heat Transfer and Fluid Mechanics
Istanbul, Turkey, August 16 - 17, 2018. https://waset.org/conference/2018/08/istanbul/ICHTFM

CFD SIMULATION OF TURBULENT CONVECTIVE HEAT TRANSFER IN RECTANGULAR MINI-CHANNELS FOR ROCKET COOLING APPLICATIONS

Dr. O. Anwar Bég, Mr. Armghan Zubair, Miss Sireetorn Kuharat & Dr. Meisam Babie

Abstract

Heat transfer is one of the most critical aspects of the rocket propulsion design process. According to released heat, thermal loads are extremely large, and thermal insulation is frequently necessary in the motor combustion chambers and nozzles.

In high temperature conditions, large thermal dilatations are present, and also the motor's parts mechanical characteristics decreases. These occurrences are very important in the motor design process, and they are directly dependent from them temperature field. This is the reason why precise heat transfer calculation is necessary.

Non-eroding metallic throat inserts made with pure tungsten, tungsten-rhenium alloys, and tungsten-rhenium alloys doped with hafnium carbide are now common. Combustion gas temperatures can rise up to 3000 Celsius. Very high heat transfer rates from hot gases to the chamber wall must be designed for. Important research areas related to heat transfer of rocket nozzle include the internal and external heat transfer coefficient predictions, metal temperature distribution, wall cooling methods, and ceramic coatings among others.

Life extension of the nozzle, which consists of an expensive super alloy, is very effective for reduction of the running costs of a power generation plant. Accordingly, it is very important for the life assessment of the nozzle to predict the operating conditions and to establish a basis for the criteria of repair. In order to assess the life of the nozzle accurately, it is necessary to estimate its temperature distribution by prediction of the thermal environment. A cooling system is essential therefore in order to maintain engine integrity.

Methodology

ANSYS FLUENT CFD single-phase, two-dimensional turbulent forced convection simulations. We have used the data provided by Forrest[1]. The fluid enters the rectangular mini-channel with a hydraulic diameter (Dhyd) of 3.79mm. Since the experiment considers turbulent flow, a Reynold's number of 50,433 and Prandtl number of 3.0 is used as, it corroborates with the experimental values Forrest (2014) was able to collect. **Fig 1** shows the top section labelled as the isothermal length of 88.9mm along the channel, held at 333.15K. After the 88.9mm location from the datum (bottom of the channel) to 339.7mm, the section is labelled as the heated wall with a constant heat flux of 241.66 KW/m2. From 339.7mm to the 428.6mm section is considered isothermal. The aspect ratio of the channel is very high (28.5:1), and here only 2D simulations are considered.

Here we deploy the realizable k - ε model available in ANSYS FLUENT. This turbulence model is one of the most popular used in the aerospace industry since it does not impact too heavily on computational power and can accommodate quite complex geometries and also heat transfer. The purpose of using k – ε is to develop a suitable eddy viscosity formulation and eddy dissipation equation. The Reynolds averaging model is used to be able to determine the governing RANS equations and the two model equations to solve the kinetic energy 'k' and the dissipation 'ε'. Hence, the model takes the following form for the turbulent kinetic energy:

$$\frac{Dk}{\partial t} = P_k - \varepsilon + \frac{\partial}{\partial x_i}\left(\frac{V_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right)$$

$$\frac{D\varepsilon}{\partial t} = \frac{\varepsilon}{K}(C_{\varepsilon 1}P_k - C_{\varepsilon 2}\varepsilon) + \frac{\partial}{\partial x_i}\left(\frac{V_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right)$$

$$v_t = C_\mu \frac{k^2}{\varepsilon}$$

$$C_{\varepsilon 1} = 1.44,\ C_{\varepsilon 2} = 1.92,\ C_\mu = 0.09, \sigma_\varepsilon = 1.3 \text{ and } \sigma_k = 1.0$$

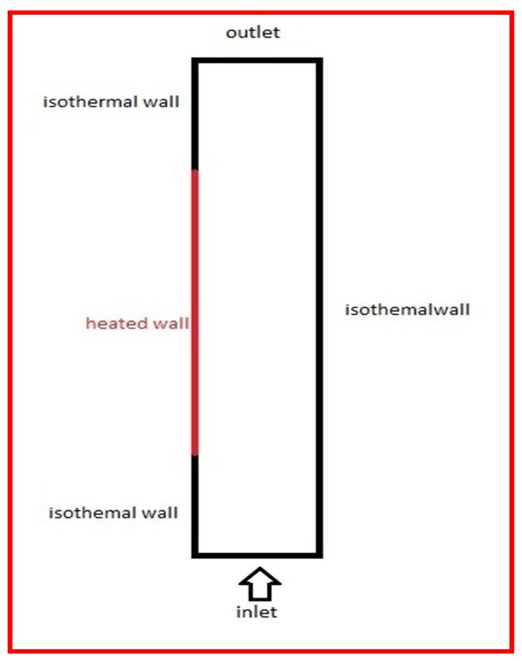


Fig.1

The CFD analysis is used to generate representative pressure, velocity and thermal fields. This grid refinement is conducted in the solver phase of the simulation to confirm adequate accuracy. The solver is set to include the double precision option to allow a higher accuracy and the parallel processing option is enabled to utilize the power of the multi-core system and the double GPU feature.

Validation

A mesh convergence study is given later to assess the optimum mesh density to collect accurate results. Hence, for this study an element size of 0.05mm was used to generate 579,120 number of elements to generate a turbulent flow model problem. Deploying a greater bias factor would increase the mesh density to the furthest edges of the channel which would prove to be useful if the focus of the study was just on a single side of the wall. Since a bulk temperature is involved with the calculations, it is essential to ensure a suitable bias factor is used to ensure the reliability of the results. Hence, this study we have opted to use a bias factor of 5 to allow greater mesh density at both edges of the channel – see below in **Fig.2,3**:

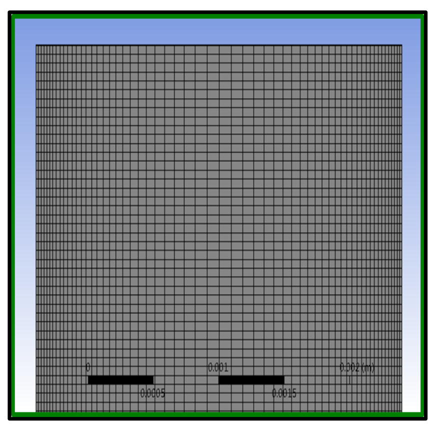


Fig. 2

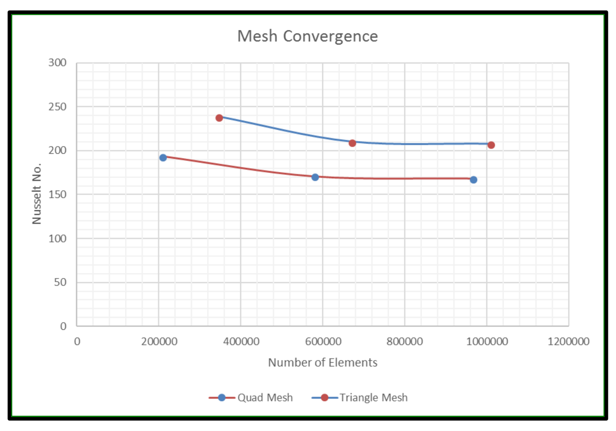


Fig. 3

For case 1 of Quad mesh at 0.07 mm cell size, 206850 elements were produced whereas in Triangle Mesh 344750 elements were produced. For the cell size 0.05 mm, in case 2, 579120 and 669150 elements were created for Quad and triangle mesh, respectively. In case 3, at 0.03 mm cell size, 965200 and 1008390 elements were generated in the Quad and Triangle mesh, respectively.

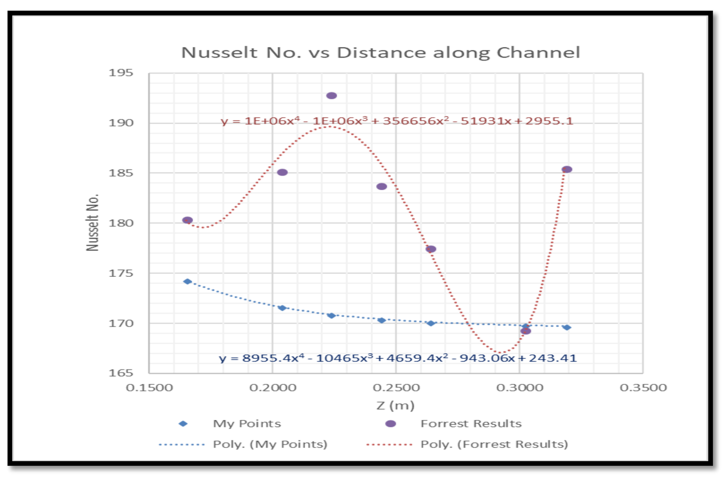


Fig. 4

To further validate the CFD computation, we compare the ANSYS FLUENT results with a graph from Forrest's data as the trend line used provides an approximate trend to his results- see **Fig. 4**. Using the R-squared function in Excel, it provides an estimate of how accurate the trend line is in relation to Forrest's results that were collected. The R-squared function was $R^2 = 0.9423$ and hence suggests that the trend line is an accurate representation with a small margin of error present. This implies that the current model can be used as a suitable baseline to design multiple Aspect Ratios.

Results

Presented below is the Aspect Ratio Study:

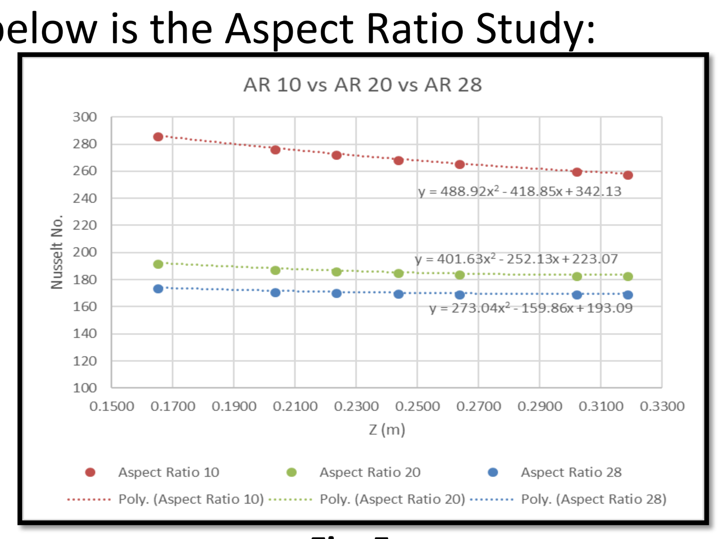


Fig. 5

A clear trend can be seen in **Fig. 5** where a larger aspect ratio appears to provide greater Nusselt number characteristics across the same channel length. Both AR 20 and AR 28 follow the gradient however aspect ratio 20:1 provides a more suitable channel that will perform better cooling characteristics than the current channel.

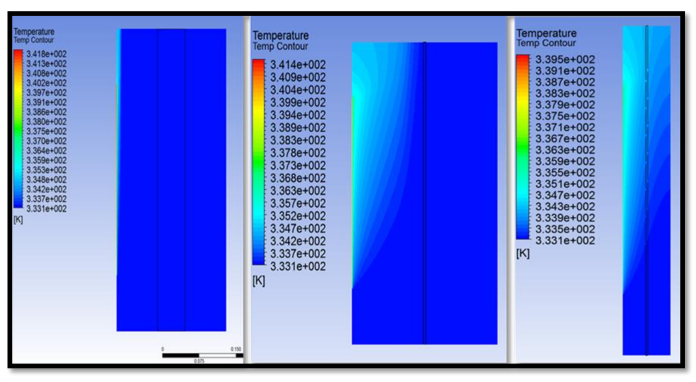
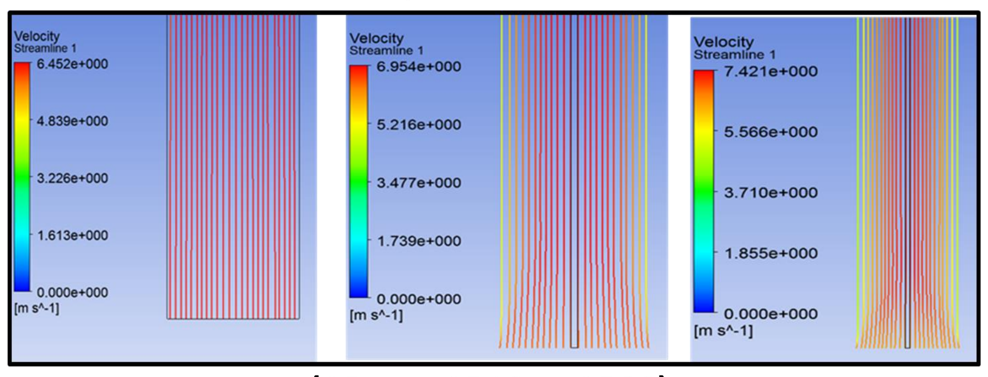


Fig. 6

Ideally the cooling channel should be capable of transferring heat towards the opposite side of the wall, which will enable a faster dissipation of heat away from the system as the pressure drop across the channel (and the velocity) will ensure a swift transfer of heat out of the system. The smallest aspect ratio appears to absorb the smallest amount of heat but achieves feeding a larger capacity of fluid in comparison to the other aspect ratios. It is also evident that thermal boundary layer thickness also shows a correlation with an increase in aspect ratio, where the layer becomes thicker. A thicker thermal boundary layer would imply that the channel has greater convective heat transfer properties. Also the maximum temperatures reached for the channels show that AR1 attains 341.8K, AR10 reaches 341.4K and AR20 peaks with a temperature of 339.5K which suggests that the maximum temperature reached is reduced with higher aspect ratios (three plots in **Fig. 6**) i.e. better cooling is attained.

Velocity Streamline Plots



(AR 1 vs AR10 vs AR20)
Fig. 7

Fig. 7 visualizes the velocity streamlines for different aspect ratios. Since the model have no slip boundaries at either sides of the wall, this suggests that maximum friction will be applied to the walls, which may influence the flow of the fluid through the channel. For the case of AR1, the effect of no-slip boundary conditions has a marginal effect to the core flow through the square-duct channel and hence there is virtually minimal deceleration/acceleration experienced by the flow. However, the cases of AR10 and AR 20 show the effect of the no-slip boundary as it influences the core flow through the channel and hence causes the flow to develop ***after entry*** in the channel. This is also indicated by the curvature of the streamlines in x where at the inlet AR20 has a greater curvature of the streamlines in comparison to AR10 and AR1. **This implies that high aspect ratios are more likely to experience fully developed flow through the channels.**

Conclusions

This research sought to conduct an investigation into the turbulent flow in mini-channels to observe the reliability of Forrest's data, in order to evaluate the cooling performance at different aspect ratios. A successful was model developed in **ANSYS FLUENT** with a mean average error of 5.97% relative to Forrest's data collected at a Reynold's number 50,443 with a Prandtl number of 3.01. This suggests that the simulation model created for turbulent flow was suitable to set as a foundation for the study of different aspect ratios in the channel.

Multiple aspect ratios were also considered to understand the influence of high aspect ratios to analyse the best performing cooling channel, which was determined to be the highest aspect ratio channels. Hence, the ~28:1 aspect ratio provided the best characteristics to ensure effective cooling. However the limitations on mesh density and hardware have curtailed the sophistication achievable for the turbulence characteristics. LES and DNS could not be used, nor could the RNG FLUENT turbulence model. Also only linear rectangular channels were considered, i.e. curvature was ignored. Furthermore we only considered conventional water coolant.

From this CFD study the variation of aspect ratio provided a deeper appreciation of the effect of small to high aspect ratios with regard to cooling channels. Hence, when considering an application for the channel, the geometry of the aspect ratio must play a crucial role in optimizing cooling performance.

Further extensions to this study could include the use of **nano-particle doping** to achieve better cooling efficiency by modifying the coolant thermal conductivity, viscosity etc. This constitutes a good pathway for future MSc and possible PhD studies building on the work reported here.

Contact

Dr. O. Anwar Bég & Miss Sireetorn Kuharat
Department of Aeronautical and Mechanical Engineering, University of Salford, Newton Building, Manchester, M5 4WT, UK.
Email: *S.Kuharat@edu.salford.ac.uk* **&** *O.A.Beg@salford.ac.uk*
Mr. Armghan Zubair
Tenencia Aerospace Design, Dakota House, Coventry Airport, Coventry, CV8 3AZ, UK.
Email: *armghanz@gmail.com*
Dr. Meisam Babaie
Department of Petroleum and Gas Engineering, University of Salford, UK.
Email: *m.babaie@salford.ac.uk*

References

1. Forrest, E. C., Study of Turbulent Single-Phase Heat Transfer and Onset of Nucleate Boiling in High Aspect Ratio Mini-Channels, Cambridge: Massachusetts Institute of Technology (MIT) 2014.
2. Forrest, E. C., Hu, L., Buongiorno, J. & McKrell, T. J., Convective Heat Transfer in a High Aspect. Journal of Heat Transfer, 16, 10, 2015.
3. Han, L. S., Hydrodynamic Entrance Lengths for Incompressible Laminar Flow in Rectangular Ducts. Journals of Applied Mechanics, Issue 27, pp. 403-409, 1960.
4. Winter, F. H., Rockets into Space. Massachusetts: Harvard University Press, USA, 1990.
5. Wang, Q., Wu, F., Zeng, M., Luo, L., Sun, J.: Numerical simulation and optimization on heat transfer and fluid flow in cooling channel of liquid rocket engine thrust chamber. International Journal for computer aided Engineering and Software 23, 907–921, 2006.
6. Naraghi, M.H., Dunn, S., Coats, D.: A Model for Design and Analysis of Regeneratively Cooled Rocket Engines. AIAA 2004-03852, 2004.
7. A. Zubair, CFD Simulation of rocket channel cooling, MSc Dissertation, Aerospace Engineering, University of Salford, UK, February, 2017.
8. S. Kuharat, *Simulation of Convective Heat Transfer in Mini Channels*, Manchester: University of Salford, 2017.